NUMERICAL SIMULATIONS ON THE EFFECTS OF USING VENTILATION FAN ON CONDITIONS OF AIR INSIDE A CAR PASSENGER COMPARTMENT

INTAN SABARIAH BINTI SABRI

A thesis submitted in fulfilment of the
requirements for the award of the degree of
Master of Engineering (Mechanical)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

MARCH 2014

To my beloved...
father, Sabri bin Abdullah
mother, Rebeah bt Hj. Harun
brothers, sisters-in-law,
nieces and my nephews.

ACKNOWLEDGEMENTS

In the name of Allah, the Most Gracious and the Most Merciful

First and foremost, I would like to express my deepest gratitude to Allah swt for giving me an opportunity, strength and His guidance to complete this research on time. With His blessings may this work be beneficial for the whole of mankind.

I am deeply indebted to my supervisor, Dr. Haslinda bt Mohamed Kamar for her constant encouragement, guidance, understanding, motivation as well as valuable advice throughout the completion of this research. My special thanks extended to my co-supervisor, Associate Professor Dr Nazri bin Kamsah for his support, patience and guidance throughout this research. He has taught me his professionalism and the profound art of research which inevitably is reflected in this thesis.

I am indebted to University Teknologi Malaysia (UTM) and Ministry of Higher Education (MOHE) for funding this Master's study as well as Research Management Centre (RMC), UTM (Vot No. 00G41) for financial support.

My utmost appreciation goes to my beloved parents and siblings for their love, understanding and endlessly praying for my success. My appreciation also extended to my dearest friends for their moral support and friendship. To my greatest friend, Nik Nabilah Nik Mohd Naser, thank you for your supports and patience throughout this friendship. May Allah bless all of us.

ABSTRACT

The use of a mechanical ventilation fan is one strategy that can be employed to reduce high air temperature build up in a car passenger compartment when it is parked in the sun. This study developed a numerical simulation to investigate the effects of using a ventilation fan on the air temperature inside the passenger compartment when it is parked in the open space during a sunny day. A computational fluid dynamics (CFD) method was used to perform the numerical simulations. FLUENT software was employed to develop the passenger compartment model and simulate the air flow conditions inside the passenger compartment. The validation of the numerical simulation was done by comparing the air temperatures from the CFD simulation results against the air temperatures at two selected points obtained from the field measurement from 12 pm to 3 pm. On average, the air temperature prediction at the front and rear compartment show good agreement with the measured data, with a difference of about 2.5% and 1.6%, respectively. The validated numerical simulation was used to perform a parametric study to investigate the effects of the ventilation fan location, number of the ventilation fans as well as the air velocity at the ventilation fans, on the air temperature and air flow pattern inside the passenger compartment. A three-dimensional (3D) steady–state simulation results show that placing one ventilation fan at the rear deck with an air velocity of 2.84 m/s reduces the air temperature at the front and rear compartment by 4°C. Placing four ventilation fans at the roof lowers the air temperatures at the front and rear compartments by 6°C and 7°C, respectively. Increasing the air velocity from 2.84 m/s to 15.67 m/s at four ventilation fans placed at the roof suggests the highest reduction of the air temperature inside the passenger compartment by 8°C.

ABSTRAK

Penggunaan kipas pengudaraan mekanikal merupakan satu strategi yang boleh digunakan untuk mengurangkan suhu udara yang tinggi di dalam ruang penumpang sebuah kereta apabila ianya diletakkan di bawah sinaran matahari. Kajian ini membangunkan simulasi berangka untuk menyiasat kesan–kesan penggunaan kipas pengudaraan terhadap suhu udara di dalam ruang penumpang sebuah kereta apabila ianya diletakkan di kawasan terbuka pada hari yang panas terik. Kaedah Perkomputeran Dinamik Bendalir Berbantukan Komputer (CFD) digunakan untuk melakukan simulasi berangka. Perisian FLUENT digunakan untuk membangunkan model ruang penumpang dan mensimulasi keadaan aliran udara di dalam ruang penumpang. Pengesahan simulasi berangka telah dilakukan dengan membandingkan suhu udara daripada simulasi berangka dengan suhu udara pada dua titik yang diperolehi daripada pengukuran sebenar daripada pukul 12 tengahari hingga 3 petang. Secara purata, ramalan suhu udara pada ruang hadapan dan belakang menunjukkan persetujuan yang baik dengan data yang diukur, dengan perbezaan masing-masing adalah sebanyak 2.5% dan 1.6%. Simulasi berangka yang telah disahkan digunakan untuk melaksanakan kajian parametrik untuk menyiasat kesan-kesan lokasi kipas pengudaraan, bilangan kipas pengudaraan dan juga halaju udara pada kipas pengudaraan, ke atas suhu udara dan corak aliran udara di dalam ruang penumpang. Keputusan simulasi keadaan mantap tiga dimensi (3D) menunjukkan bahawa meletakkan satu kipas pengudaraan pada dek belakang dengan halaju udara 2.84 m/s mengurangkan suhu udara pada ruang hadapan dan belakang sebanyak 4°C. Meletakkan empat kipas pengudaraan pada bumbung menurunkan suhu udara pada ruang hadapan dan belakang masing-masing sebanyak 6°C dan 7°C. Peningkatan halaju udara pada empat kipas pengudaraan yang diletakkan pada bumbung daripada 2.84 m/s kepada 15.67 m/s menunjukkan pengurangan suhu udara tertinggi di dalam ruang penumpang iaitu sebanyak 8°C.